CFD simulations on marine burner flames

The marine industry is changing with new demands concerning high energy efficiency, fuel flexibility and lower emissions of NOX and SOX. A collaboration between the company Alfa Laval and Technical University of Denmark has been established to support the development of the next generation of marine burners. The resulting auxiliary boilers shall be compact and able to operate with different fuel types, while reducing NOX emissions.

The specific boiler object of this study uses a swirl stabilized liquid fuel burner, with a pressure swirl spill-return atomizer (Fig.1). The combustion chamber is enclosed in a water jacket used for water heating and evaporation, and a convective heat exchanger at the furnace outlet super-heats the steam.

The purpose of the present study is to gather detailed knowledge about the influence of fuel spray conditions on marine utility boiler flames. The main goal of work presented in this paper was to obtain a spray description to setup a particle injection region in the CFD simulations of the boiler.